# Aerothermal Technologies and Design Tools for Advanced HP Turbines

**C. Brisset, J.M. Duboué, P. Picot**
Turbine Aero and Cooling Department
SNECMA MOTEURS
77550 Moissy Cramayel, France

**Keywords** : *Turbine, cooling circuits, CFD*

## Abstract

The design of advanced turbines has to meet ever more ambitious requirements. Higher performances and component life must be reached within shorter design cycles at lower cost. Snecma Moteurs is building up an extensive experience in the design and manufacturing of the next generation of high-pressure turbines for military and commercial aircraft engines.
The purpose of the present contribution is to report on the recent advances in the design of cooling technologies for High Pressure (HP) turbine blades.
First of all, the new concepts of advanced cooling circuits are described and their interests in terms of life duration put into relief.
Several examples demonstrate the way these new advanced cooling technologies can be optimized by carrying out high-tech Computational Fluid Dynamics (CFD) methods.
Finally, validation cases of the aerothermal methodology analysis built up at Snecma Moteurs are detailed. They present the interest of being focused on engine tests results and show the degree of performance and quality that Snecma Moteurs HP turbines aerothermal technology and design tools have now reached.

## 1. Introduction

For the new generation of aircraft engines, the development and design of advanced cooling technologies for high-pressure turbines are a key issue for most of the critical engine related factors, such as fuel consumption, component life, weight, and development costs. The high temperature levels required by this new generation of engine demand to improve the efficiency of cooling systems for HP turbine blades. For many years, an important effort has been devoted at Snecma Moteurs to designing, analyzing, industrializing and manufacturing advanced cooling technologies. To achieve this challenge, the design and analysis is supported by accurate CFD tools for the prediction of aerothermal phenomena [1].
The purpose of the present contribution is to report on the recent advances in the design of cooling technologies for HP blades.
The characteristics of the new cooling concepts are presented ; their interest is put into relief for both performances and life duration. For these advanced blades, the improvement is based on recent progress made in casting and drilling.
Then, modeling is described : to achieve life duration target, turbine designers need to control the very complex flow phenomena occurring not only around the blade but also inside a rotating advanced cooling circuit. In this way, the use of CFD tools in the day-to-day design methodology has been intensively put in place. CFD tools produce boundary conditions for a Finite Element conduction solver that provides the thermal fields in the material. For both internal and external flows, the prediction capacity of the heat transfer tools used at Snecma Moteurs will be detailed through several examples. Especially, their use for the cooling improvement of blade critical area is demonstrated. For that purpose, the simulation of technological effects is widely taken into account in both internal and external CFD calculations : blade platforms, blade tip clearance, film cooling holes, internal blade channel with ribs… At Snecma Moteurs, a strong interaction between the design team and the CFD integration team has always allowed an early use of advanced methods in the design process.

*Paper presented at the RTO AVT Symposium on “Advanced Flow Management: Part B – Heat Transfer and Cooling in Propulsion and Power Systems”, held in Loen, Norway, 7-11 May 2001, and published in RTO-MP-069(I).*

# Report Documentation Page

*Form Approved*
*OMB No. 0704-0188*



| | | |
|---|---|---|
| 1. REPORT DATE<br>**00 MAR 2003** | 2. REPORT TYPE<br>**N/A** | 3. DATES COVERED<br>**-** |

| | |
|---|---|
| 4. TITLE AND SUBTITLE<br>**Aerothermal Technologies and Design Tools for Advanced HP Turbines** | 5a. CONTRACT NUMBER<br>5b. GRANT NUMBER<br>5c. PROGRAM ELEMENT NUMBER |
| 6. AUTHOR(S) | 5d. PROJECT NUMBER<br>5e. TASK NUMBER<br>5f. WORK UNIT NUMBER |
| 7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)<br>**NATO Research and Technology Organisation BP 25, 7 Rue Ancelle, F-92201 Neuilly-Sue-Seine Cedex, France** | 8. PERFORMING ORGANIZATION REPORT NUMBER |
| 9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) | 10. SPONSOR/MONITOR'S ACRONYM(S)<br>11. SPONSOR/MONITOR'S REPORT NUMBER(S) |

12. DISTRIBUTION/AVAILABILITY STATEMENT
**Approved for public release, distribution unlimited**

13. SUPPLEMENTARY NOTES
**Also see ADM001490, presented at RTO Applied Vehicle Technology Panel (AVT)Symposium held in Leon, Norway on 7-11 May 2001, The original document contains color images.**

14. ABSTRACT

15. SUBJECT TERMS

| 16. SECURITY CLASSIFICATION OF: | | | 17. LIMITATION OF ABSTRACT | 18. NUMBER OF PAGES | 19a. NAME OF RESPONSIBLE PERSON |
|---|---|---|---|---|---|
| a. REPORT<br>**unclassified** | b. ABSTRACT<br>**unclassified** | c. THIS PAGE<br>**unclassified** | **UU** | **18** | |

**Standard Form 298 (Rev. 8-98)**
Prescribed by ANSI Std Z39-18

Finally, this paper will focus on the experimental validation of the prediction models. A constant effort is dedicated to the comparison, validation and calibration of methods. This means in particular that heavily instrumented rigs, representative of real engine flows are used to produce an appropriate validation data base. In this contribution, we will rather detail several "engine" validation cases like comparison between aerothermal predictions and thermosensitive paints results.

## 2. Characteristics of new cooling concepts

For more than a decade from now, important studies have been focused on cooling advanced technologies. The development programs have involved simultaneously aerodynamics and cooling designers, stress and life specialists, and casting and drilling manufacturers in order to lead toward a thermally and mechanically efficient solution which would be feasible with reasonable costs.
Blade cooling improvements has been reached by optimizing :

- internal convection by increasing heat transfer coefficients and exchanges surfaces (wall cooling concept),
- drilling configurations for increasing film effectiveness on the external surface of the blade,
- Thermal Barrier Coating with reduced conductivity.

### 2.1 Optimization of internal convection

Two generations of advanced cooling circuits for HP turbine blades will be detailed hereafter. Both are "wall cooling" technologies; their characteristics in terms of cooling effectiveness and life duration will be compared to the current basic on multipass cavities blade cooling for which an example is shown on **figure 1**.

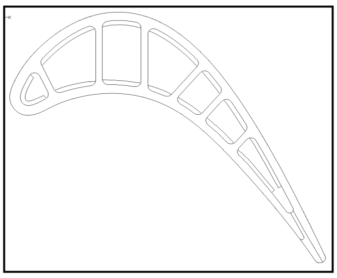


**Figure n°1 : Multipass cavities blade cooling**

These new cooling concepts enable to run at high HP turbine inlet temperature levels with reasonable cooling air amounts (less than 5% of the HP compressor entry mass-flow). All the concepts described in the present contribution are patented.

The first advanced wall cooling circuit developed by Snecma Moteurs, named **"advanced cooling circuit 1"** hereafter, is based on serpentines with turbulated radial cavities which present a high length to width ratio. A scheme of this typical circuit is presented on **figure 2**. Two serpentines are located in the central part of the blade respectively along the suction and pressure surfaces. Turbulators are facing the external envelope of the blade. A wide central cavity is set between the two serpentines The leading edge is cooled through impingement holes located in the partition wall between the leading edge and the central cavity. The air used for the impingement takes the benefit of being protected from hot gas by the two serpentines which avoid its overheating along the wall. The rear part of the blade is cooled with a current concept : multipass cavities and ejection of the cooling air in the trailing edge.

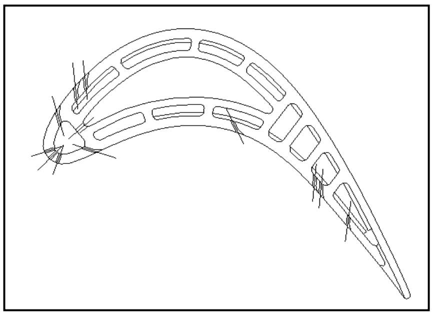


**Figure n°2 : Snecma Moteurs 1st advanced wall cooling circuit**

The feasibility of this first advanced wall cooling circuit of wall cooling circuit has been demonstrated. Its thermal efficiency has been demonstrated on a HP spool at high inlet turbine temperature (above 2000K). This kind of cooling circuit is ready to be introduced on an engine application.

The next generation of Snecma moteurs advanced wall cooling circuits is more prospective than the previous one. This technology is called **"advanced cooling circuit 2"**. The main part of the blade excepted the leading edge and the trailing edge is cooled by very small width radial cavities with

pedestrals inside. Under the groove, the air is send into a large central cavity before being ejected through cooling holes. A scheme of this typical circuit is shown on **figure 3**.

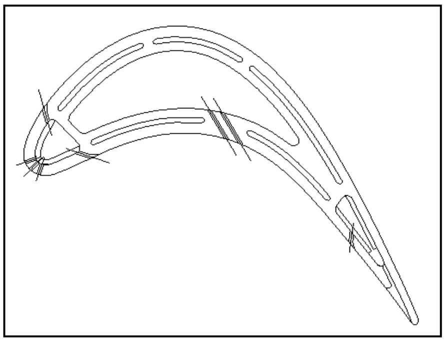


**Figure n°3 : Snecma Moteurs 2nd advanced wall cooling circuit**

A new casting process has been developed to realize this advanced cooling concept. The feasibility has been demonstrated.

Advanced cooling systems allow an engine to run at higher inlet turbine temperature levels. But the interest of such cooling technologies can be also put into relief with current military engine temperature levels by both reducing cooling air mass-flow and increasing life duration. Low cooling air mass-flow enable sensitive fuel consumption improvements.

The following table presents, for the same given cooling mass-flow and aerodynamic blade envelope and with identical running thermodynamic conditions, a comparison between averaged cooling effectiveness (**figure 4**) and life duration. The circuits are coated with the same thermal barrier coating. Life duration takes into account both creep and oligocyclic fatigue damages.

| Cooling circuits | Current multipass | Advanced wall cooling 1 | Advanced wall cooling 2 |
|---|---|---|---|
| **Total number of cavities** | 8 | 13 | 8 |
| **effectiveness** | Datum | **X 1.1** | **X 1.18** |
| **Life durations** | Datum | **X 2** | **X 3** |

These figures demonstrate the important potential of Snecma Moteur wall cooling circuits which would first enhance significantly the HP turbine life duration of current engines and which are secondly needed for the next generation of aircraft engines.

The second advanced wall cooling circuit presents a great potential, but its application is more linked to the development of new fighter engines running at very high inlet temperature.

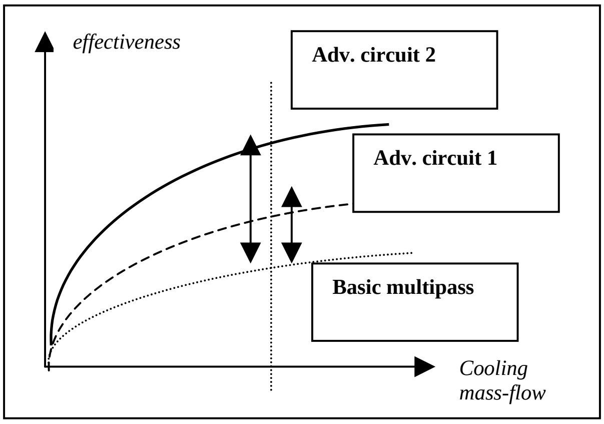



**Figure n°4 : Comparison of thermal effectiveness for a given mass-flow**

The definition of the thermal effectiveness is detailed hereafter :

$$\eta = \frac{\mathbf{Tg - Tw}}{\mathbf{Tg - Tc}} \qquad \text{with :}$$

**η :** Cooling effectiveness

**Tw** : Averaged metal temperature

**Tg :** Datum hot Gas temperature

**Tc :** Cooling air feed temperature

## 2.2 Optimization of film cooling

Film cooling optimization has been studied through coolant ejection angle influence on external heat transfer (see scheme on **figure 5**). To increase both film cooling effectiveness and reducing local metal temperature close to the cooling holes, the coolant ejection angle has been reduced. Three values of ejection angle has been studied for an HP exit guide vane. One row was located on the pressure side and the other on the suction side in a high acceleration zone (see **figure 6**). The exit Mach number was closed to 1,05.

In parallel, a feasibility program has been set up to achieve the drilling of these low ejection angle holes.

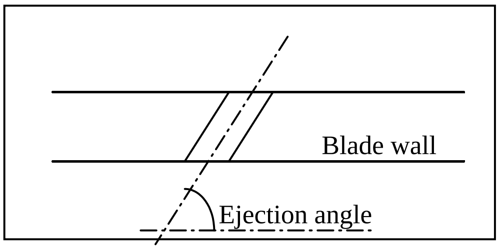



**Figure n°5 : Film cooling geometry**

The analysis is based on 3D Navier-Stokes simulations for which the injection of coolant in the freestream is much detailed, the mesh being ever refined at the injection locations. These calculations produce both heat transfer coefficient and gas temperature which are used as boundary conditions for a Finite Element conduction model.

The resulting metal temperatures on the external envelope for each ejection angles (see **figure 7**) clearly show that low ejection angles enable to locally reduce metal temperature. An optimum is reached with an ejection angle which matches the drilling capacities of the manufacturing shop.

More detailed results of this calculation will be reported in the following paragraph dedicated to the importance of CFD methods in the design and analysis of turbine cooling systems, especially for advanced concepts.

## 3. Power of analysis and prediction CFD tools

Numerical tools represent a significant means to improve the design process of turbines, which leads to higher performances, cost and cycle saving as well as to lower associated risks.

The intensive use of powerful CFD methods in the development of advanced cooling technologies aims to better optimize the turbine blade life. Accurate predictions in terms of heat transfer and gas temperature are produced. They lead to a metal temperature field which will be used to determine elastic-viscous-plastic (EVP) stresses in order to determine the damage of HP blades. It will be illustrated through two actual cases.

The first one presents an analysis of external heat flux due to film cooling ejection on a blade surface with three values of ejection angle.

The second example concerns a 3D viscous simulation of the internal flow of an advanced cooling circuit.

### 3.1 CFD tools description

The external and internal flow phenomena are both calculated through 3D viscous codes, named CANARI for the external aerodynamic and thermal predictions and MSD (Mathilda Saphir Diamant) for the simulations into blades cavities, which have been developed in close cooperation with ONERA, the National French Research Organism for Aeronautics and Space.

The aerothermal blade to blade simulation package includes an in-house mesh and pre-processing generator, the ONERA code CANARI and a post-processing based on Iris-Explorer tool.
The CANARI code is a compressible, finite volume, time marching , multi-domain code solving full Navier-Stokes equations on a structured grid [2], [3]. The numerical core is based on a four step Runge-Kutta explicit scheme combined to Jameson and Turkel second and fourth order numerical dissipation model. On top of this scheme, residuals are smoothed by an implicit technique. This code is optimized for transonic flows.
Although capable of, the code does not use a wall law approach for steady computation ; therefore fine grids are required for the computations. For each row, the equations are formulated in a conservative manner and are solved in its own frame of reference.

The CFD methodology for the investigation of internal flows uses the HEXA mesh generator from ICEM Technologies, an in-house interactive pre-processing, and the ONERA code named MSD [4], [5]. This code is able to cover a wide range of flows, in particular the low speed flows; it solves the 3D unsteady Reynolds-averaged compressible Navier-Stokes equations for a mixing of perfect gases. It uses a finite volume technique on curvilinear structured multi-domain grid. The time integration is implicit with first order accuracy. The spatial discretization scheme is second order accurate. The convection terms are evaluated through a flux difference splitting Roe scheme. Several turbulence models are available in the MSD code, from basic algebraic model to algebraic stress model. The standard model used for internal cavity

simulations is the two-equation model k-l. Depending on mesh thickness at the wall, the code switches from a wall function mode to a low-Reynolds mode.

The procedure to obtain a three dimensional 3D blade metal temperature field uses the HEXA/TETRA/PRISM mesh generators from ICEM Technologies, an in-house interactive pre-processing, the commercial conduction solver ABAQUS and the post-processing tool ABAQUS/Viewer. The pre-processing manages all the boundary conditions whatever the way these ones are calculated : 3D Navier-Stokes calculations or 1D correlation based code. The commercial thermal solver ABAQUS uses classical iso-parametric finite elements and handle 3D turbine blade applications with hexahedral and/or tetrahedral elements.

### 3.2 Film cooling calculations

For a given aerodynamic flow-field, the number and location of row and holes, the pitch and shape of holes for each row, the streamwise and compound injection angles of holes are the most important geometrical parameters for the design of a film cooled blade. The day to day film cooling analysis is performed with a 1D in-house correlation based code, whose results are superimposed on the 3D viscous CANARI uncooled blade results. In parallel to this methodology, a refined analysis has been carried out with a 3D Navier-Stokes code including film cooling modeling. This advanced 3D method, able to combine much more physical phenomena compared to a 1D based method, is useful to quantify the effect of coolant ejection angle on aerothermal performances.

The code used for the film cooling prediction is the 3D Navier-Stokes CANARI ; a sophisticated chimera technique with overlapping meshes is adopted to model the aerothermal flow-field environment with a good accuracy in particular within the blade boundary layer development [6], [7].

The example presented in the previous section is detailed hereafter. Three values of ejection angle have been simulated for two row of holes : the first one is located on the pressure side and the second one on the suction side. To limit calculation running times, only a slice of the blade (20% of the height of the blade), located at blade mid-height section, has been simulated. A view of the mesh, composed of about 650000 points, is presented on **figure 8.**

The heat flux on the external surface of the blade is the result of these simulations; the resulting heat transfer coefficient is plotted chordwisely for each configuration on **figure 9**. The static temperature profiles in the boundary layer just upstream the ejection holes are shown for each case for both suction side and pressure side on **figures 10 and 11** .

It can be firstly noticed that on the suction side, the ejection is located in a high acceleration area, where the aerodynamic and thermal boundary layers are very thin, the ejection of coolant flow reduces static temperature close to the wall which leads to minimize temperature gradient and reduce locally external heat transfer coefficients. The lower the ejection angle is, the more reduced the static temperature is, in the vicinity of the wall . The heat transfer coefficients remains low, 40% lower versus the datum configuration until 75% of the axial chord.

The behavior on the pressure side is very different. For the datum configuration, the cooling flow creates a local separation which makes the local heat coefficient increase ; the cooling air doesn't reduce the static temperature close to the wall.

By getting the ejection angle at a lower value, the coolant flow is kept closer to the wall surface and make the static temperature, and therefore the heat transfer flux decrease. The calculations permit to determine an optimum angle.

Through this detailed example, the importance of advanced CFD methods has been demonstrated for both optimizing blade cooling, with the best configuration for film cooling holes, and for finding an optimum value for the ejection angle in relation with the drilling capability.

### 3.3 Calculation of the internal flow of an advanced cooling circuit

For day-to-day internal cooling analysis, a 1D correlation based code is used, taking into account the whole geometrical and aerothermal parameters which may influence the internal heat transfer : cavity aspect ratio, rib passage aspect ratio, rib blockage ratio, rib inclination, rib pitch to height ratio, pedestral blockage

ratio, Reynolds number….These in-house correlation are based on the extensive Snecma Moteurs previous experience.
For complex geometries, 3D Navier-Stokes calculations with the MSD code are carried out for detail analysis of local flows and heat transfers.
Current 1D codes are not so accurate as to analyze flows in small sized radial cavities like the serpentine of the first advanced wall cooling circuit presented in the previous paragraph. In fact this configuration is closed to the limit of the correlation validation range. That is the reason why a 3D viscous approach has been carried out on the serpentine located near the pressure side and composed of three cavities. Two of them include ribs on one side which are orthogonal to the main flow, the opposite wall being smooth.
The mesh is hexaedric and structured including an H topology for the cavity. Each rib is wrapped by an H mesh too, while the turning at the blade tip is described with an O topology. The whole mesh includes about 3600000 nodes with 320000 elements. A view is presented on **figure 12.** Boundary conditions at the entry of the serpentine are total pressure and total temperature, while mass-flow is imposed for each film cooling hole of the exit. At convergence the differences between inlet mass-flow and exit mass-flow of the serpentine are less than 0.1%.
The main aerodynamic parameters of the flow field given by the 3D Navier-Stokes computation display a good agreement with those issue from the 1D "day-to-day" approach : the differences in terms of coolant air heating and total pressure losses are lower than 5%.
A local analysis of the relative Mach number in the whole serpentine (**figure 13**) enables to see two separation areas in the turning of the circuit which make an obstruction. The most interesting knowledge in terms of cooling design is the flow field around the ribs : the Mach number is the lowest between two ribs and the highest on the smooth wall. This is due to the rib obstruction in a small sized cavity. This phenomenon is also highlighted on **figure 14 ;** its consequences on the predicted thermal wall temperature, based on flux calculation, is an increase of 10° on 15% of span. In spite of this phenomenon, the concept keeps all its interest.

This second example shows clearly that powerful CFD methods enable to investigate in detail complex aerothermal problems, as for globally and locally thermal predictions.

## 4. Validation

To validate an HP blade design, several steps need to be taken into account. First of all, a constant effort is to be dedicated to the comparison, validation and calibration of methods. This means in particular that heavily instrumented rigs representative of real engine flows be used to produce an appropriate validation database [8]. These experimental investigations with Snecma Moteurs associated Research institutes, equipped with a comprehensive and high quality set of measurements are available at Snecma Moteurs to validate and calibrate advanced numerical methods. But, the validation of a cooled blade design go further than the validation of CFD methods through basic phenomena. The whole methodology involved in the design and analysis process must be validated too, through the behavior of blades operating in engine conditions. Engine tests with turbines blade coated with appropriate thermal sensitive paints give a thermal field on the external blade surface. Metallurgic analysis are carried out after an endurance engine test to estimate local metal temperatures which has been produced in running conditions. Two examples based on these last two means of validation are developed hereafter.

The first one presents a comparison between the temperature field at the surface of a high pressure cooled blade issued from a thermal sensitive paints test run on an engine and the prediction. The blade is wrapped with a Thermal Barrier Coating made of zirconia.
The predicted surface temperatures are the final achievement of the aerothermal methodology analysis. The 3D temperature field is provided by the conduction solver ABAQUS. The thermal barrier coating is treated with shell elements in this application.
External boundary conditions (heat transfer coefficients, reference temperatures) are first issued from a 3D Navier-Stokes calculation performed with the CANARI code. The measured tip clearance is modeled in this 3D viscous calculation whose boundary conditions come from a through flow model in which leakage flows are simulated in terms of mass-flow and associated losses ; the resulting total pressure and temperature radial gradients imposed at the inlet of the blade influence the

calculated external heat flux. The film cooling analysis is performed with a 1D in-house correlation-based code, whose results are superimposed on the 3D Navier-Stokes CANARI blade results. Internal boundary conditions (heat transfer coefficients and temperature) performed by the cooling circuit are calculated through a 1D correlation-based code which takes into account all the geometrical parameters and specific features of the cooling system ; rig test measurements or local analysis results provided by the 3D Navier-Stokes code MSD may be included.
The external flow simulations based on the Navier-Stokes code CANARI and the conduction solver are used independently, a manual interaction model being used between the fluid and the solid information : wall temperatures from the conduction solver are introduced in the 3D viscous code. At Snecma Moteurs, three manual iterations between solid and fluid are done by the designers to converge to satisfactory 3D material temperature field.
**Figures 15 and 16** present a comparison between 3D temperatures predicted on the external surface of an HP rotor blade coated with TBC, and the ones obtained with thermal sensitive paints. The predicted temperatures display a very good agreement with the paints test data on both blade and platform. The influence of secondary flows on external heat flux are taken into account with good accuracy.

To illustrate the way thermal predictions are validated at Snecma Moteurs, a critical and very complex area of an HP rotor blade, the cooled groove at blade tip, has been chosen. This part is particularly sensitive, brazing operations may be necessary to close up the gap due to ceramic cores during the casting of the cooling circuit; the quality of the brazing depends on the metal temperature levels close to it. The thermal sensitive paints are difficult to achieve in this area on which metal temperatures are often high. That is the reason why metallurgical analyses have been used to validate thermal predictions in the blade groove. Micrographic expert inspections of the single crystal structure in which the blade is cast, enable to evaluate a metal temperature range during the engine running conditions.
A 3D conduction model focused on the blade tip groove has been carried out in engine running conditions with a methodology identical as the one used for the previous example. The predicted metal temperatures have been compared to the results of the expert evaluations in three planes perpendicular to the groove walls **(figure 17)**. The reasonable agreement proves that our thermal analysis methodology based on powerful CFD tools is able to take into account complex geometries.

These last two examples allow to conclude that the use of powerful CFD methods validated through engine test results after being calibrated with Research experimental data bases , enable to better optimize the life duration of turbine airfoils cooled with advanced concepts by producing satisfactory predictions in terms of metal temperature field.

## 5. Conclusion

Snecma Moteurs has built up an extensive experience in the design and manufacturing of high-pressure turbines. This will feature much higher turbine inlet temperature than current State-of-The-Art ones. It will also improve the part durability which will enhance affordability of the next generation of military and commercial aircraft engines. To achieve the longest turbine life with the minimum cooling flow, new advanced wall cooling circuits have been developed along with an important effort dedicated to ease the manufacturing (casting and drilling) of these new cooling concepts. The first advanced wall cooling circuit, shown in the present contribution, which means with a short term view for an engine application, enable to double life duration of current HP blades with reasonable costs. The second advanced wall cooling circuit described in this paper presents a most important potential in terms of cooling effectiveness, however its application is rather dedicated to very high inlet temperatures.
The design of such technologies needs advanced CFD methods which help to go ahead in the improvement and optimization of cooling concepts. The aerothermal methodology analysis built up at Snecma Moteurs, involving several CFD codes, which has been successfully validated through engines test results, guarantees the quality of predicted metal temperatures fields on which life duration calculations are based.

The described examples bring out the degree of performance and quality that Snecma Moteurs HP turbine aerothermal technology and design tools have now reached.

## 6. Acknowledgements

The authors wish to thank gratefully Snecma Moteurs Direction for the authorization to publish the present paper. The financial support of DGA (French Defense Procurement Agency) for a great part of these studies, is fully acknowledged. It must be emphasized that the results presented in that paper have been obtained by a team of engineers of the Snecma Moteurs Turbine Aero and Cooling Department.

## References

[1] Duboué, J-M. ; Liamis, N ; Paté, L (2000) : Recent Advanced in Aerothermal Turbine Design and Analysis 36th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit 17-19 July 2000 Huntsville, Alabama. Paper AIAA 2000-3355.

[2] Liamis, N & Couaillier, V. (1994) : Unsteady Euler and Navier-Stokes Flow Simulations with an Implicit Runge-Kutta Method. Proceeding of the second ECCOMAS Computational Fluid Dynamics Conference, 5-8 September 1994, Stuttgart, Germany, John Wiley & Sons1994.

[3] Vuillot, A-M.; Couaillier, V.; Liamis, N. (1993) : 3-D Turbomachinery Euler and Navier-Stokes Calculations with a Multidomain Cell-Centered Approach. AIAA Paper 93-2576

[4] Dutoya, D. & Errera M.P. (1991) : Le code MATHILDA. Modèles physiques, réseau de calcul et methode numérique. Rapport technique ONERA n°42/3473EN

[5] Paté, L. & Duboué, J-M. (1998) : CFD : A Tool to Design Jet Engine Cooling Systems AIAA Paper 98-3563

[6] Fougères, J-M. & Heider, R. (1994) : Three Dimensional Navier-Stokes Prediction of Heat Transfer with Film Cooling. ASME Paper 94-GT-14

[7] Ginibre, P. ; Lefebvre, M. ; Liamis, N. : Numerical Investigation of Heat Transfer on Film Cooled Turbine Blades. ICHMT Turbine 2000 Congress, Antalya, Turkey, 2000

[8] Liamis, N.; Brisset, Ch.; Lacorre, F.; Duboué J-M. (1999) : CFD Analysis of Dual Spool Turbine Configuration. AIAA Paper 99-2525

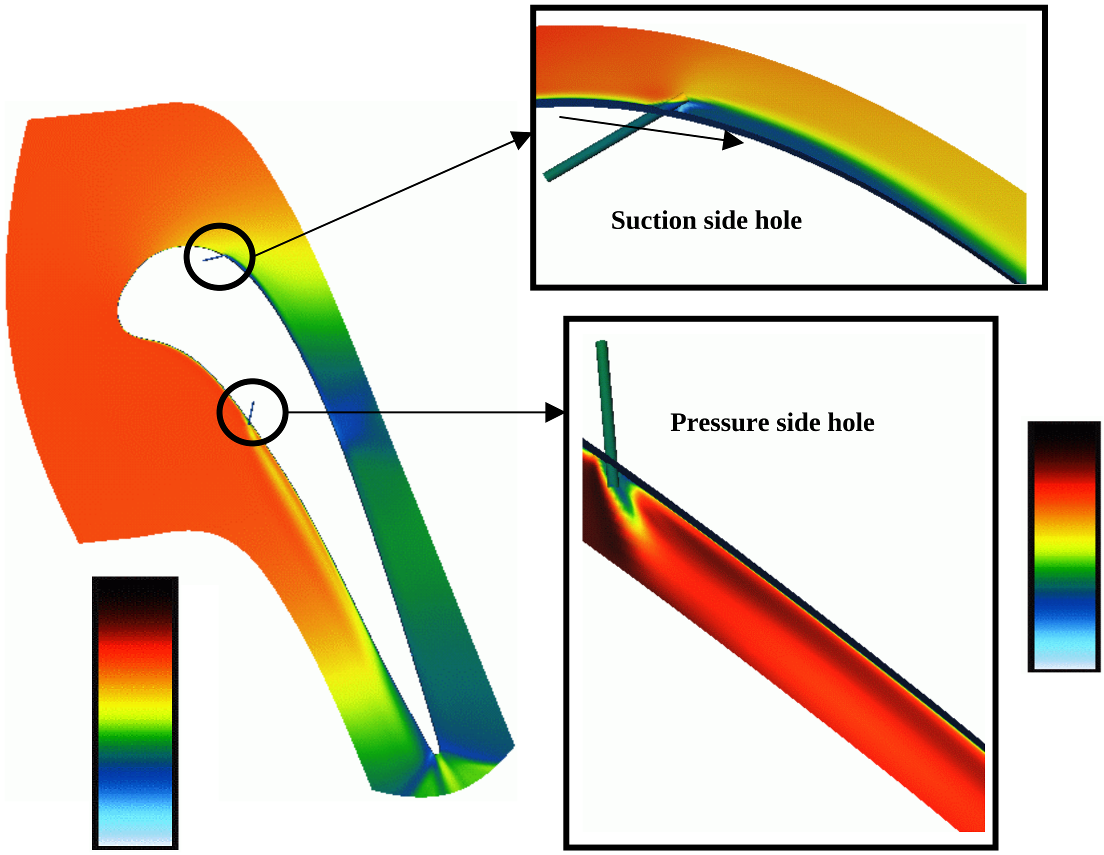



**Figure n°6** : **Film cooling calculation : Static temperature distribution**

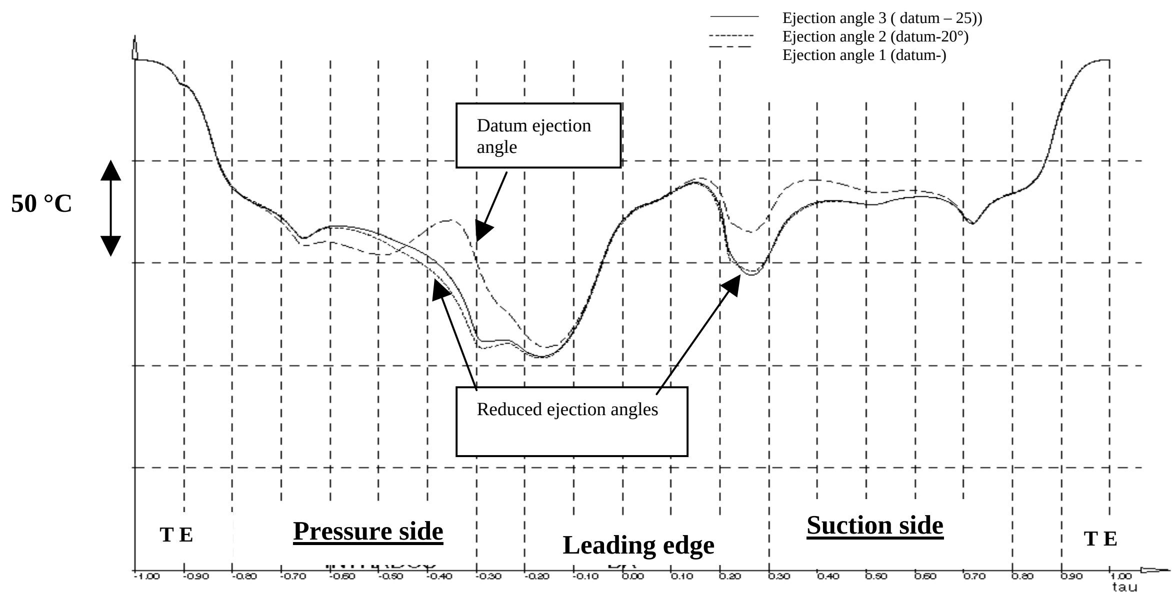



**Figure n°7** : **Ejection angle influence : wall temperature distribution on the profile surface**

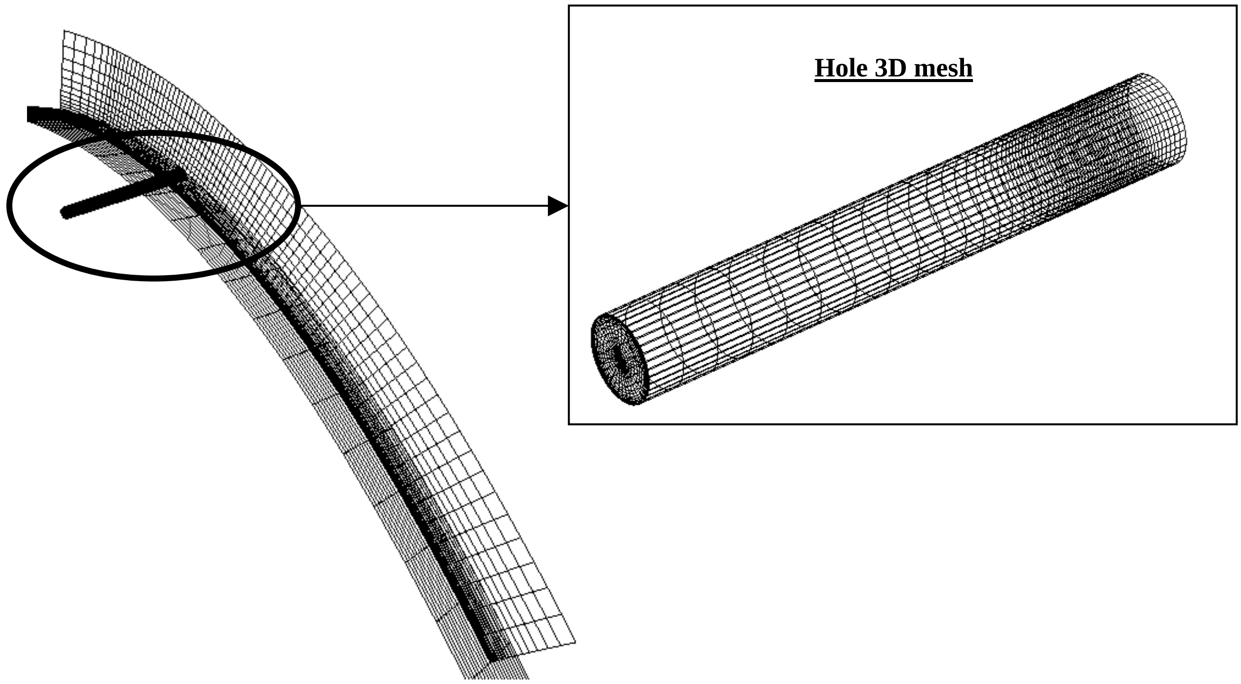



**Figure 8** : view of the film cooling calculation mesh

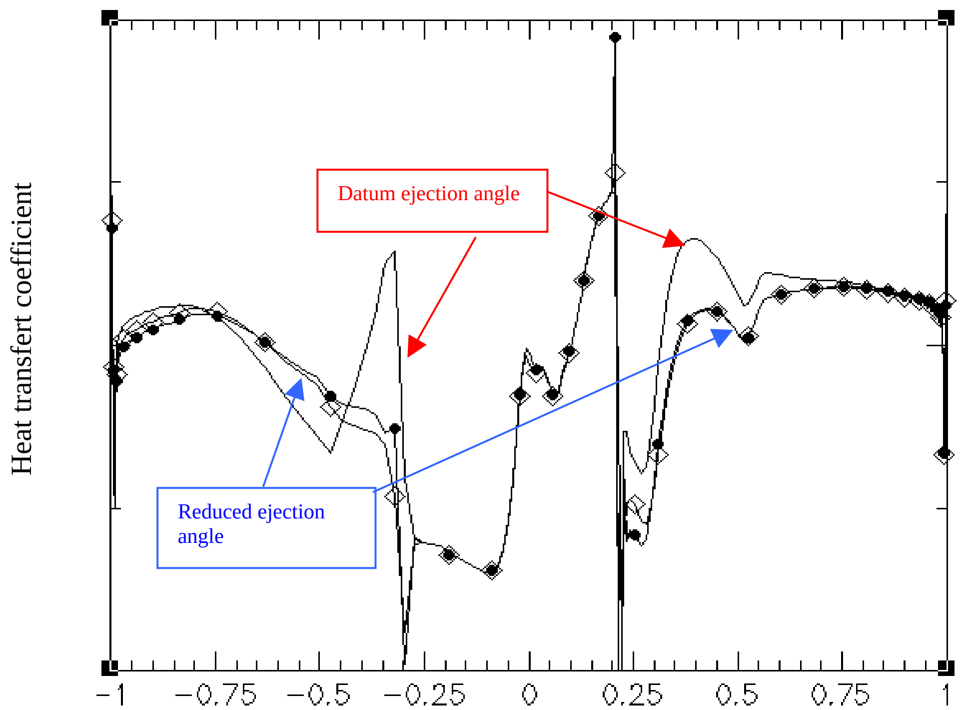



**Figure 9 : Film cooling calculation : spanwise evolutions of heat transfer coefficient for 3 values of ejection angles**

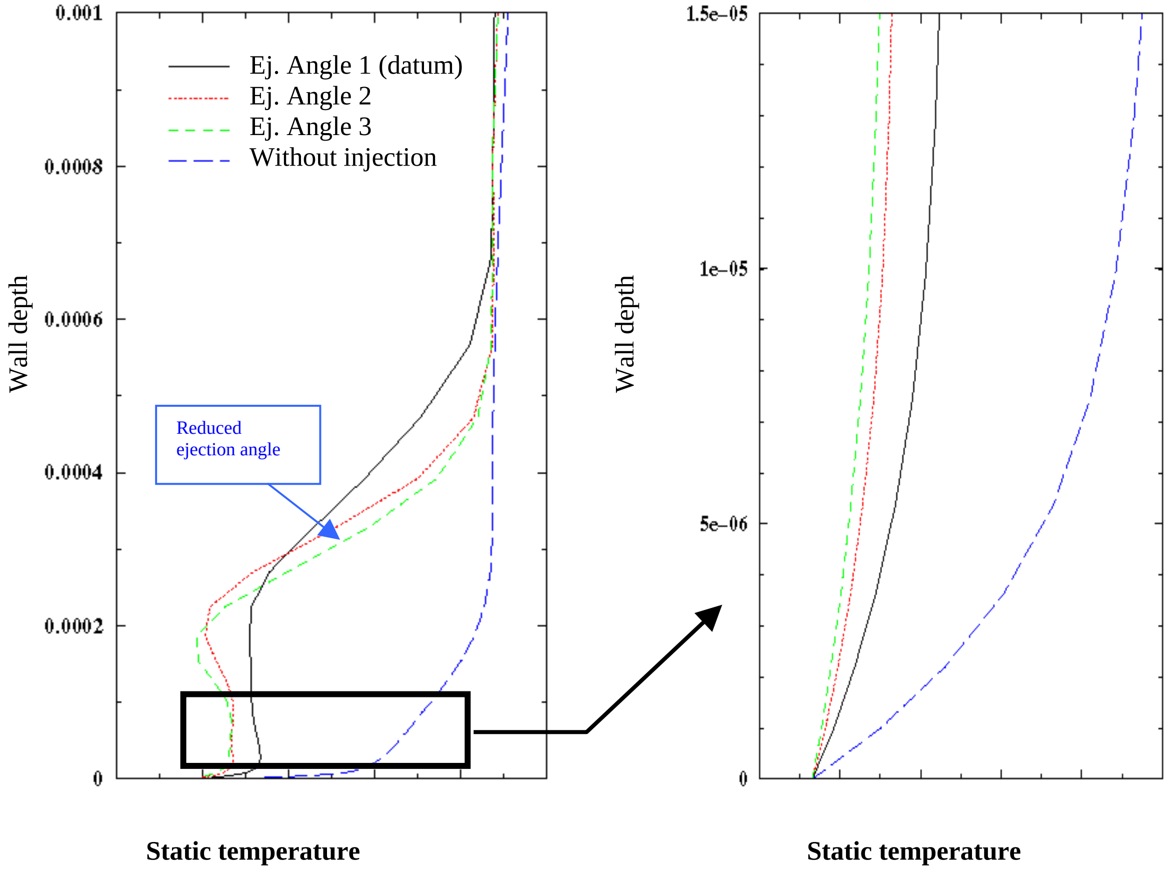



**Figure n°10** : **Temperature distribution downstream the suction side hole**

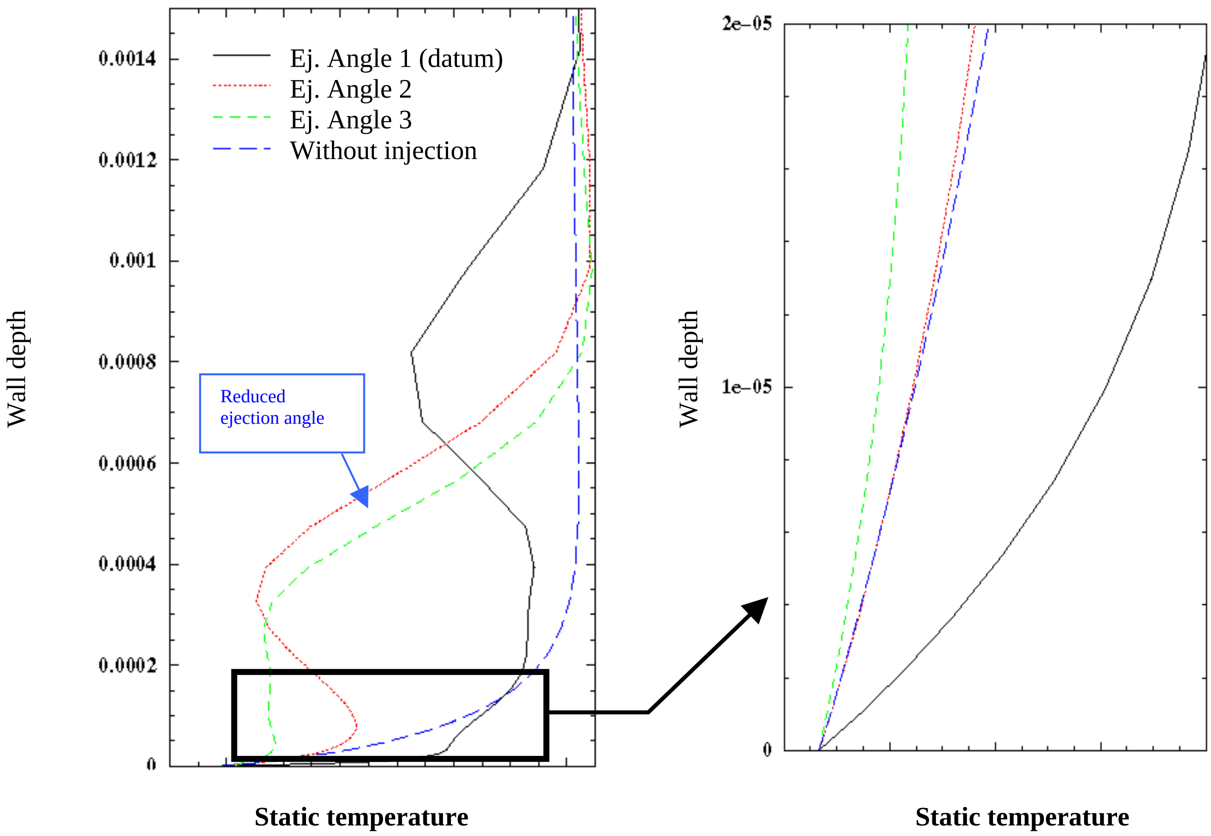



**Figure n°11** : **Temperature distribution downstream the pressure side hole**

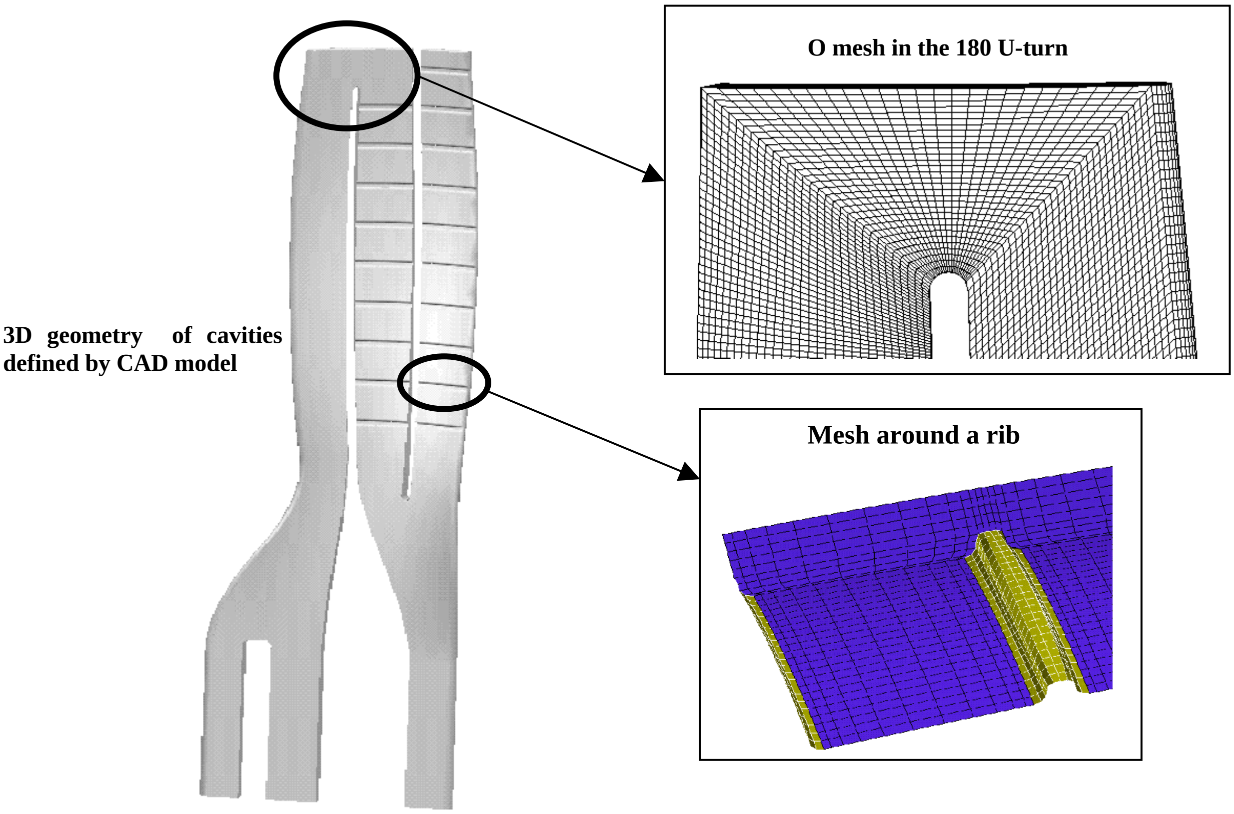



**Figure 12** : 3D viscous calculation -Mesh view of one serpentine of 1st advanced wall cooling circuit

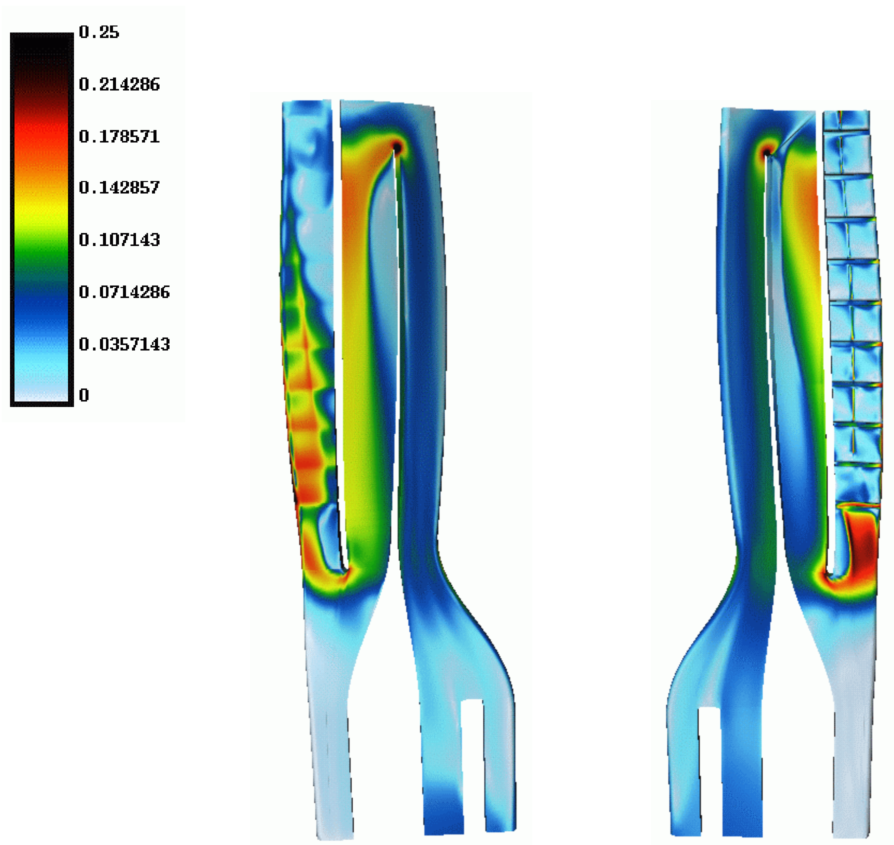
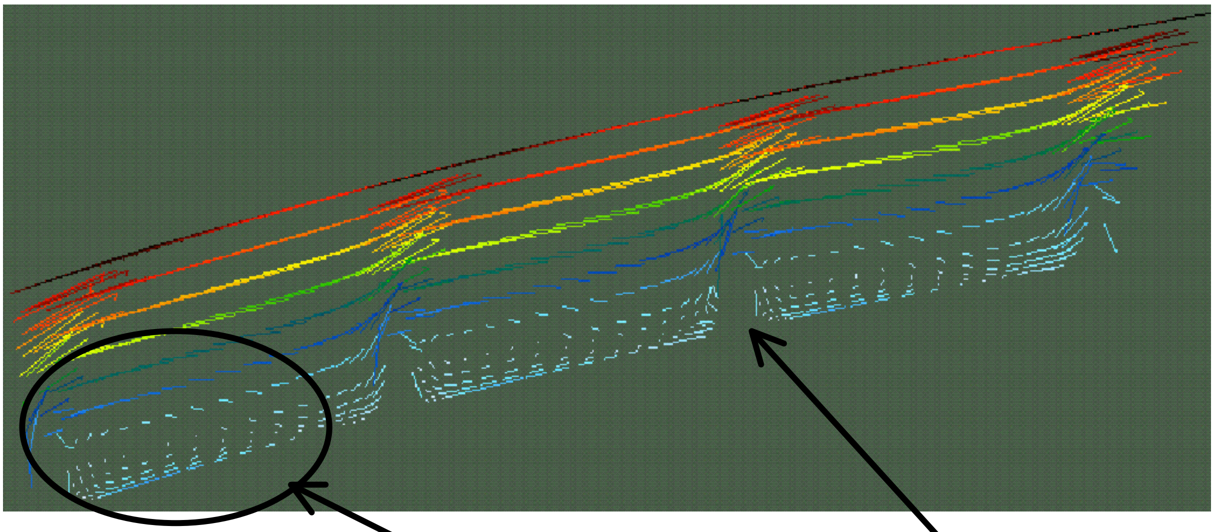
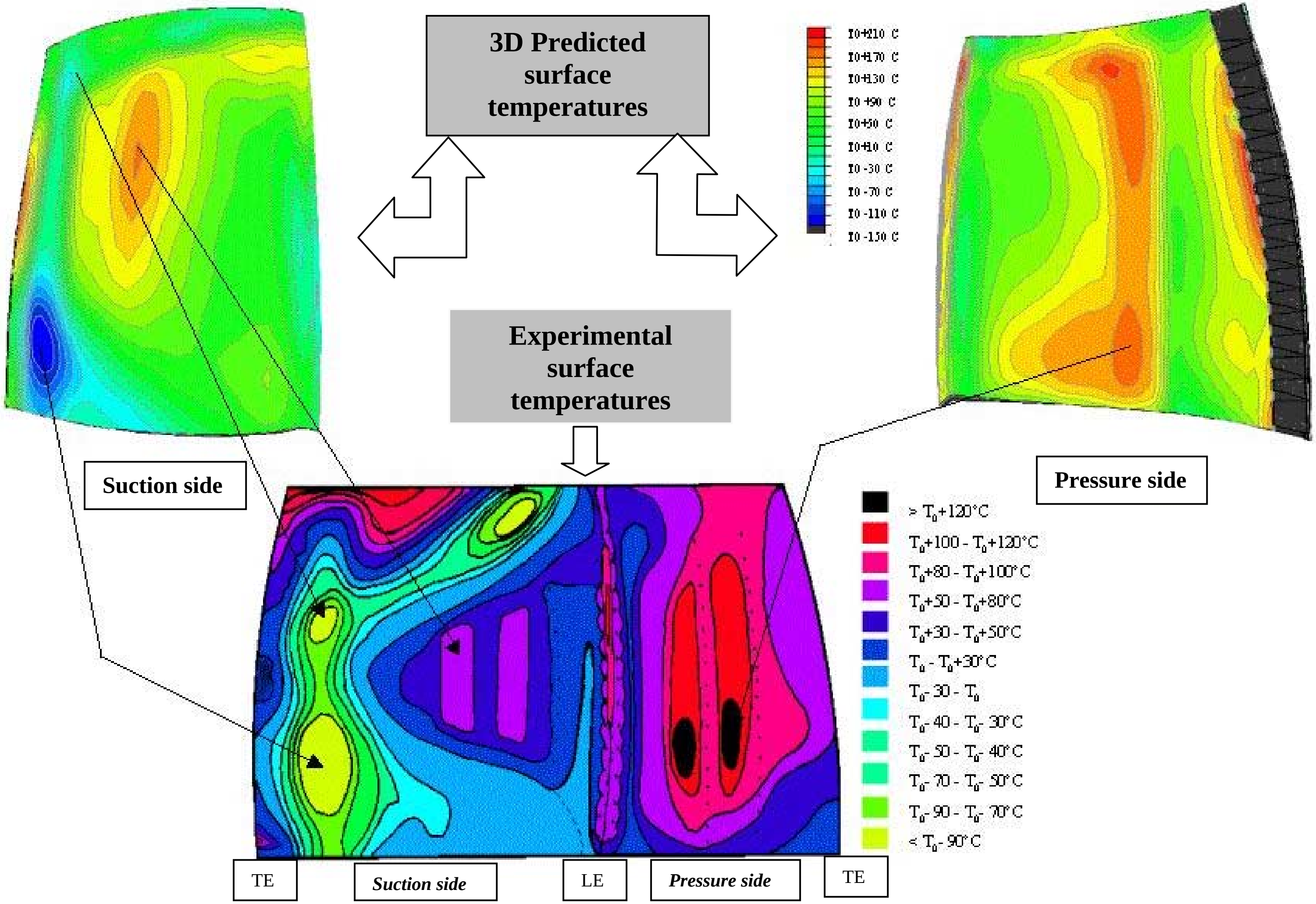


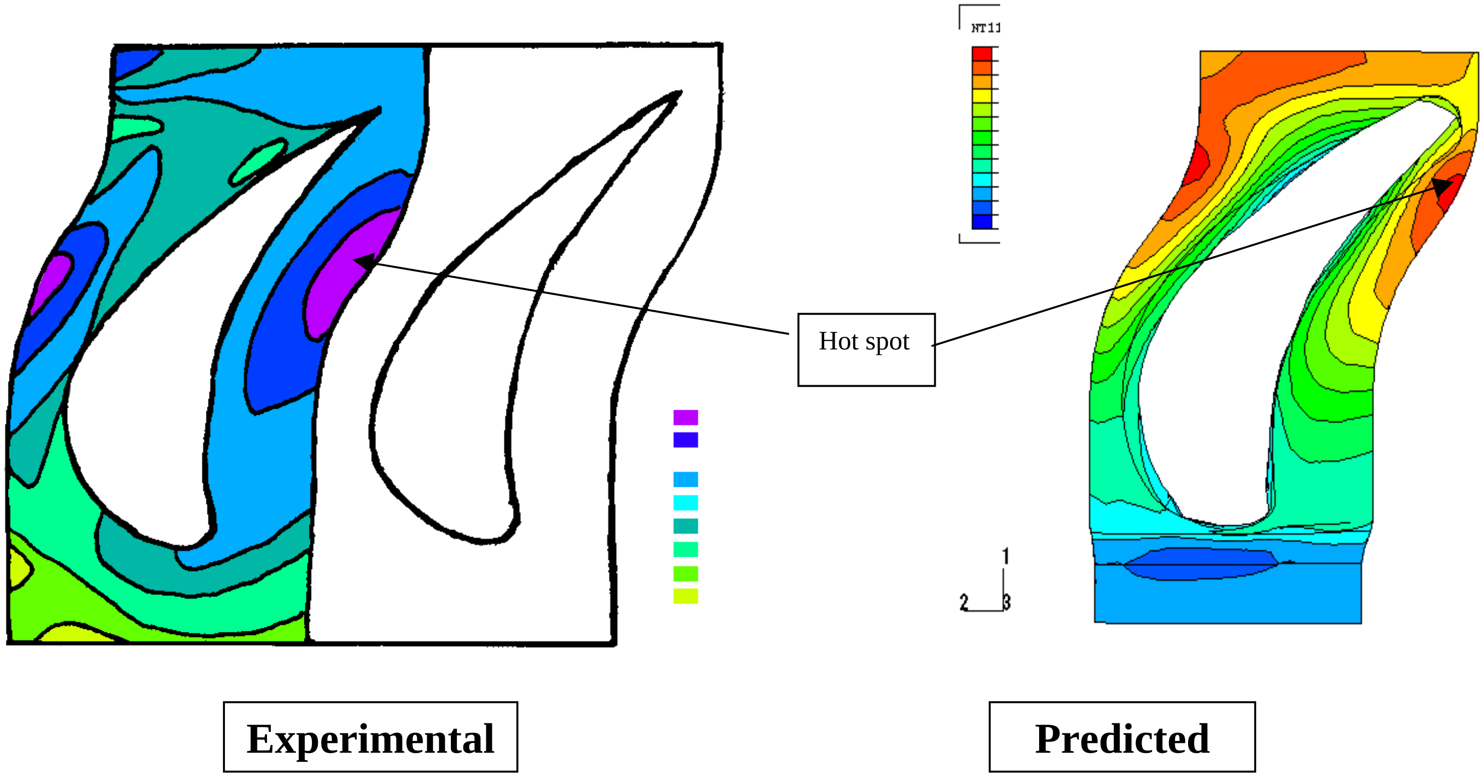

**Figure 13** : 3D viscous calculation -Relative Mach number in one serpentine of 1st advanced cooling circuit



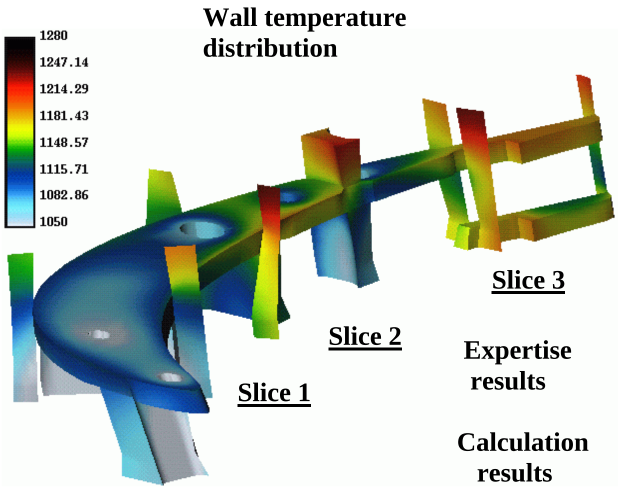

**<u>Figure 14</u> : 3D viscous calculation – flow behavior in a rib vicinity**




**<u>Figure 15</u> : comparison to thermosensitive paints**




**<u>Figure 16</u> : comparison to thermosensitive paints on platform**




| | Slice 1 | Slice 2 | Slice 3 |
|---|---|---|---|
| **Expertise results** | < 1100°C | < 1100°C | 1100→1170°C |
| **Calculation results** | 1070°C | 1090°C | 1150°C |

**Figure 17 : 3D conduction calculation results in a blade groove – comparison with metal temperature issue from metallurgic inspection**

Paper Number: 26

Name of Discusser: E. Octay, Roketsan Missile Industries Ankara, Turkey

Question:
How do you calculate heat transfer coefficient for boundary condition of the conduction solver?

Answer:
Two 3D Navier – Stokes calculations are carried out on the same mesh with CANARI code.
– the first one is adiabatic
– the second is performed with the metal temperature imposed (interactive process)

Name of Discusser: W.B. de Wolf, National Aerospace Laboratory Emmeloord, NL

Question:
In the thermal analysis you use the CANARI code without film cooling with a 1D correlation to include film cooling. Is the reason that the CANARI code for film cooling application would require too fine grids that would lead to unacceptably long computation times when applied to the full blade?

Answer:
Of course. A simulation performed with our CANARI code with all the injection holes modelled would require a very high quantity of nodes. We already performed these 3 D viscours calculations – highlighting that we are capable of.
But during a design period (short!) it can't be carried out. That's why 3D viscous calculations are performed only on a slice (≈ 20 % of the blade height) and in the "day-to-day" analysis, we used 1D correlations based code, whose results are super imposed to the results issued for the 3D viscous code CANARI without film cooling holes.
It can be noticed that with this approach the agreement with thermosensitive paint-results is very good.

Name of Discusser: Kirit Patel, Pratt & Whitney Canada

Question:
For the cooling configurations, how much reduction in flow was achieved for configuration 2 compared to datum configuration?

Answer:
If you impose for our advanced cooling circuit the same cooling mass flow as for the datum (multipass) circuit, it enable to multiply by 2 (configuration 1) by 3 (configuration 2) life duration.

But if your target is to have the same life duration between the datum configuration and the advanced wall cooling circuit Nr. 2, you can reduce the mass flow by around 1.5 or 1,7 % of high pressure compressor entry-flow.

Name of Discusser: F. Lutum, ABB Baden, Switzerland

Question:
In your calculation procedure for external flows there are two options for film cooling simulations. Could you comment on these two options used for film cooling simulations:
1) 1-D-correlation
2) 3-D-modelling

Answer:
1-D correlations are used for "day-to-day" analysis during short design cycles and are superimposed to the CANARI results (performed without film cooling)
3-D models with all the injections are possible (It has been applied on a HP rotor blade) , but computation times are too long and are not consistent with required short design period).

3-D calculations with injectionholes modelled are performed only on a slice when we consider that there may be a "failure" in the 1-D correlations based approach.

Name of Discusser: M. Elfert, DLR Cologne, Germany

Question:
You have performed flow calculations for internal and external flow domains in a separate manner. Do you have thought about to combine these to perform a calculation over the whole flow system?

Answer:
It could be done, theoretically, by "coupling" our external code (CANARI) and internal code (MSD). But in the presented paper it hasn't been taken into account. I should add that this approach is feasible, the meshes used by both solvers are structured and both codes use finite volume elements.

Name of Discusser: H.B. Weyer, DLR Cologne
Question:
How do you manage the different steps to calculate the flow in the cavity, the holes and the outside mixing area?

Answer:
With the agreement of Prof. Weyer, this question is very close to the one asked by Martin Elfert from DLR.
Thus the same answer can be given